

Solid Problem Seven

In this chapter, you will learn the following to World Class standards:

1. Sketch of Solid Problem Seven
2. Starting a 3D Part Drawing
3. Modifying How the UCS Icon is Displayed
4. Drawing a Closed Polyline Shape to be Extruded
5. Adding Circles to the Detail
6. Moving the Circle Inside the Polyline
7. Extruding Solids from a Closed Polyline and Circle
8. Subtracting a 3D Solid from Another Solid
9. Rotating a 3D Solid Using the Rotate3D Command
10. Drawing a Solid Box
11. Moving a 3D Solid
12. Constructing a Solid Perimeter Using the Line Command
13. Creating Polylines Using the Edit Polyline Tool
14. Extruding the Solid from a Closed Polyline
15. Aligning a Solid to the Master Solid
16. Subtracting 3D Solids from the Master Solid
17. Hiding the Solid
18. Moving Solid to the Origin Point
19. Saving the Solid Problem

Sketch of Solid Problem Seven

You will start the seventh solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across one line, which you can call the x-axis. In this exercise, a strategy you will continue to use is to draw closed Polylines that contain the most complex shape of the component and then use simple geometric solids to subtract detail. In problem seven, the master solid contains a chamfer drawn into the Polyline shape. You will place a circle off the edge of the main shape. After extruding the Polyline and Circle, you will subtract the cylinder. You will finish by subtracting a Box and a Wedge shape from the master solid to end with the product as shown in Figure 8.1.

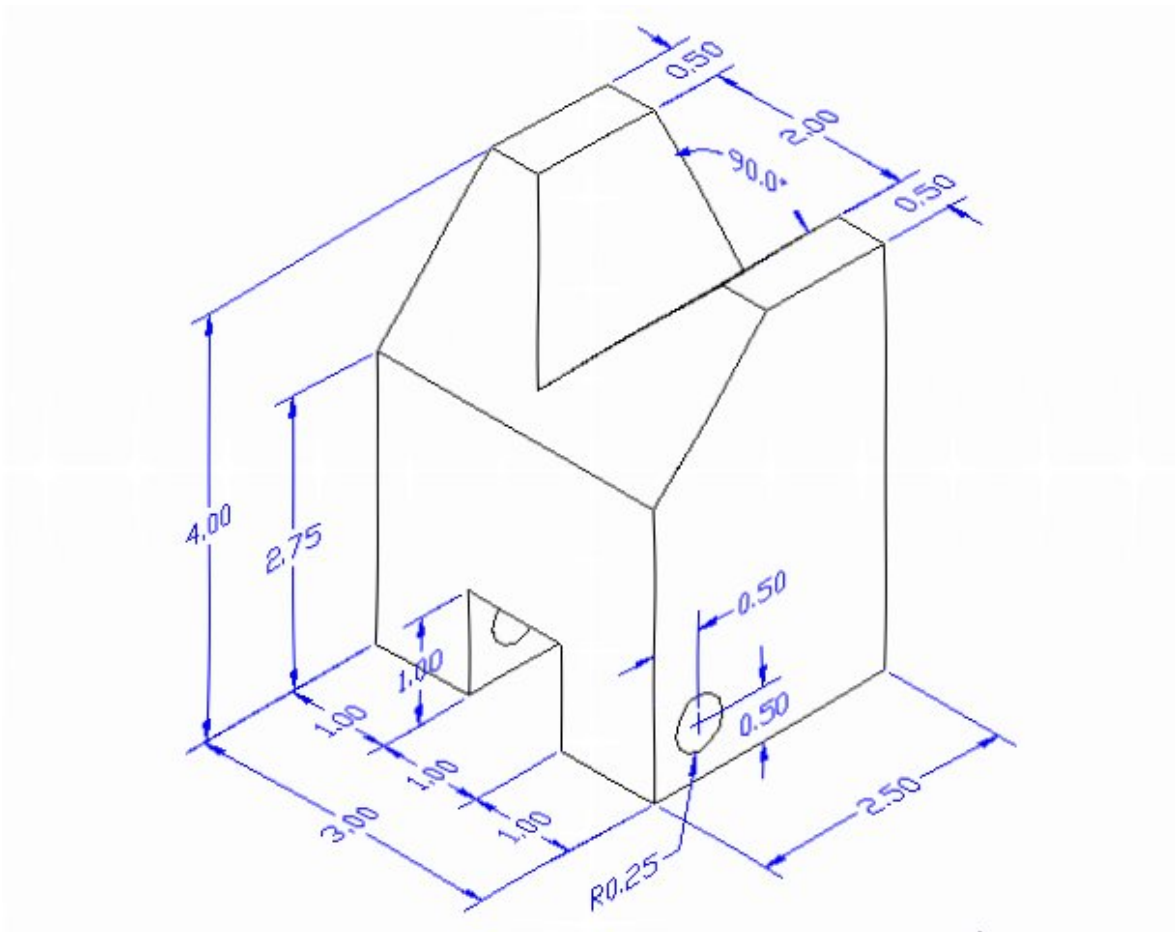


Figure 8.1 – Problem Seven Sketch

Remember, when you receive a sketch from a professional, you need to practice to quickly identify the different shapes and decide how to proceed with the solid exercise. You can see rather quickly that problem seven has different contours, so you will separate each extrusion or solid and subtract them from the master solid. This problem will help you address solid parts containing different complexities and sizes. All ten drills in the Fundamentals of 3D Drawing textbook will give you those repetitive maneuvers that will make you successful in 3D modeling.

Starting a 3D Part Drawing

In Problem Seven, you will begin the 3D drawing by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” (first button) at the top of the Start window. Choose to use English units as your Default Settings and hit OK (Figure 8.2). A new drawing file will open that contains the system variables and layer definitions which you will use later in Paper Space to finish the drawing.

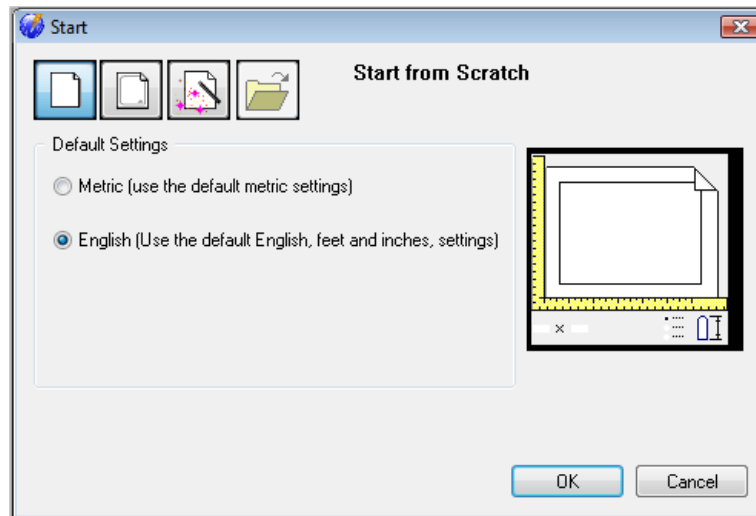
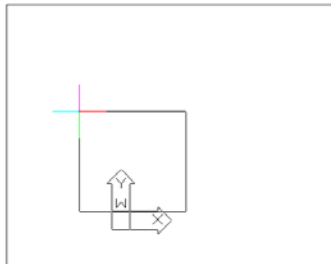


Figure 8.2 – Starting the Drawing

Modifying How the UCS Icon is Displayed



When working in progeCAD, you may notice that the UCS icon has left its position in the corner of the display. This is because the default setting places the UCS at the origin of model space if the origin is in view. Many computer aided design operators do not appreciate having additional UCS lines in with their drawing or solid, so you can change the system to show the UCS in the lower left hand corner of the display at all times.

Select Tools from the Menu Bar and then select Drawing Settings to pull up the Drawing Settings window. Under the Display tab, choose to Change settings for Display. In the UCS Icon section select the On radial button and then hit OK. Now the UCS icon will always appear in the corner of the display.

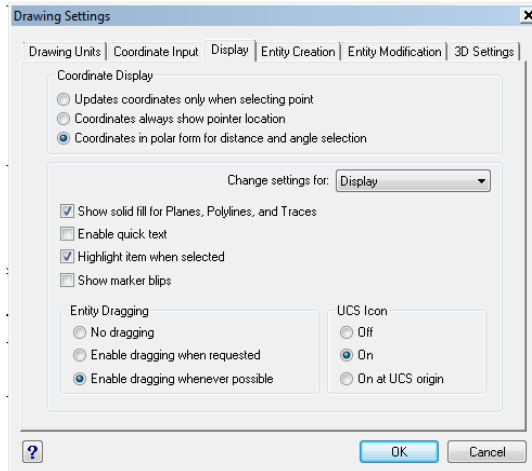


Figure 8.4 - Drawing Settings: UCS Icon

Drawing a Closed Polyline Shape to be Extruded

On the Draw Toolbar, select the Polyline tool and specify a start point in the upper left hand corner of the graphical display. In Figure 8.5, you can see the actual continuous line of data you need to specify at the command line.



Command : **_POLYLINE**

ENTER to use last point/Follow/<Start of polyline>:

Arc/Distance/Follow/Halfwidth/Width/<Next point>: **2.75**

Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **2.5**

Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **4**

Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **1.25**

Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **C**

Figure 8.5 – The Polyline Text from the Command Line

With the Ortho mode “on”, draw a **2.75** unit line downward, a **2.5** line to the right, a **4.0** line upward, a **1.25** line to the left and then type “**C**” to close the Polyline. A closed entity will appear in the graphical display. Zoom Extents to have the box fill the monitor so you can proceed to the next step in the drawing process. In Figure 8.5 you can see the step-by-step progression of building the Polyline with Figure 8.6 showing the final two-dimensional entity.

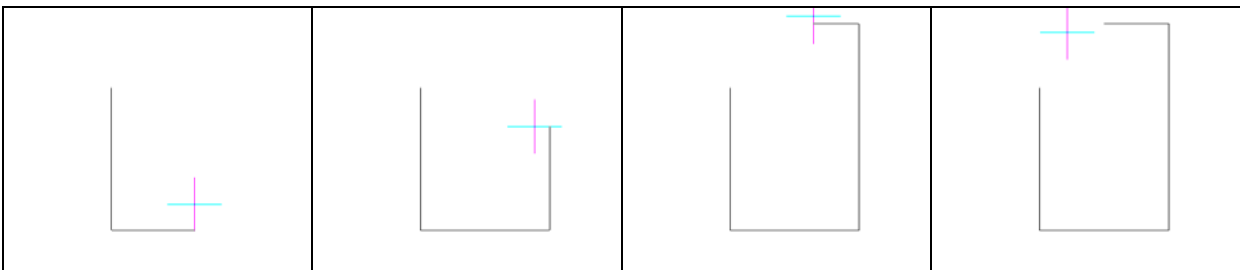
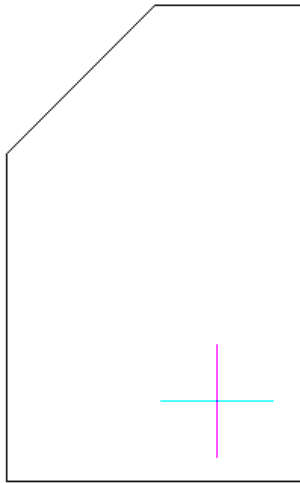


Figure 8.6 – Drawing a Series of Lines Using the Polyline Command



As you gain experience in drawing shapes in with progeCAD, notice that you can start defining the perimeter anywhere on the graphical display and not just in the lower left-hand corner. In this case, by describing the vertical and horizontal lines first, the diagonal line can be set simply by closing the polyline. When you close the perimeter, the mathematical formulation of the angled line is left to the computer. The computer will always be a faster drawer than you.

Figure 8.7 – The Finished Polyline

Adding Circles to the Detail

Next, select the Circle command on the Draw toolbar. Place the mouse aperture at the lower left Endpoint as shown in Figure 8.8 and an Endpoint Esnap symbol (square) will appear on the end of the line. Select that point as the center point of the circle by selecting with the left mouse button. Then type “**D**” when the command line prompts you to specify the “Diameter/⟨Radius⟩:” and hit ENTER. Type **0.5** as the diameter and the circle will appear on the corner as shown in Figure 8.8.

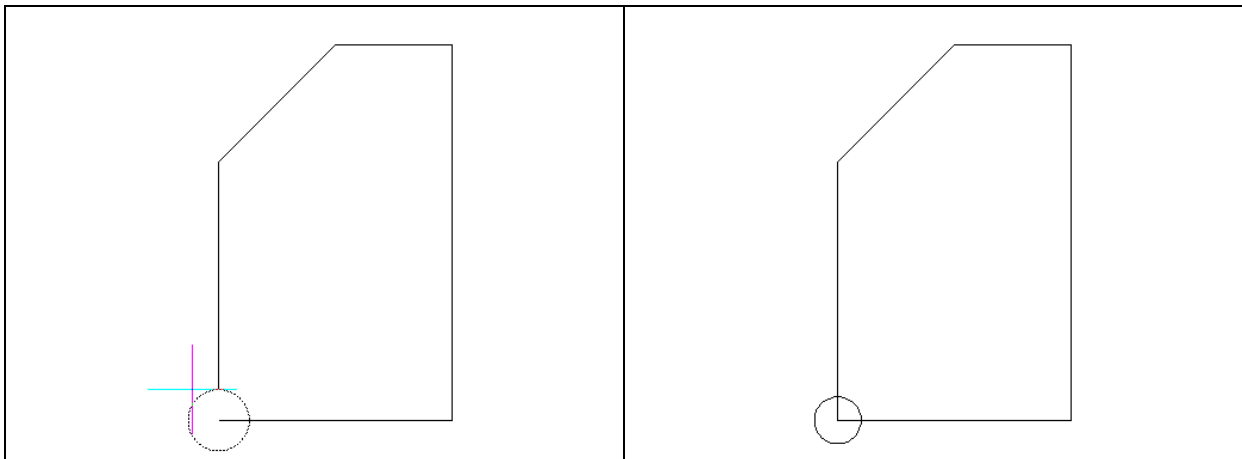
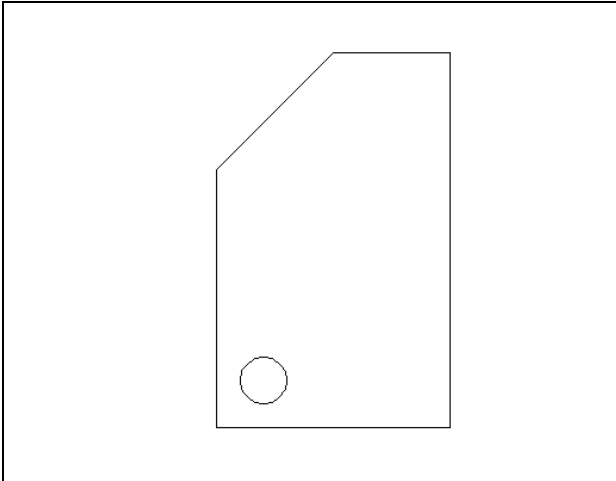


Figure 8.8 – Selecting the Center of the Circle

Figure 8.9 – The 0.5 Circle placed

Moving a Circle Inside the Polyline

Before extruding both the Polyline and the Circle, you will need to position the circle at its precise location. Use the Move tool on the Modify toolbar to accomplish the task.



Once you choose the Move command, the prompt will be “Select entities to move:”. Pick the circle and the command line will respond with “Entities in set: 1”, so hit Enter to go the second part of the command. Next, you need to “Vector/<Base point>:” on the graphical display. Pick your base point anywhere on the display. Hit **ENTER** and then type **@0.5,0.5** as the “Displacement point:”, which will move the entity a distance of 0.5 in the X-axis, 0.5 in the Y-axis, and the default of 0 units in the Z-axis. The drawing will appear as shown in Figure 8.10.

Figure 8.10 – Positioning the Circle

Extruding Solid from a Closed Polyline and Circle

Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar. You now have two shapes that are single entities.

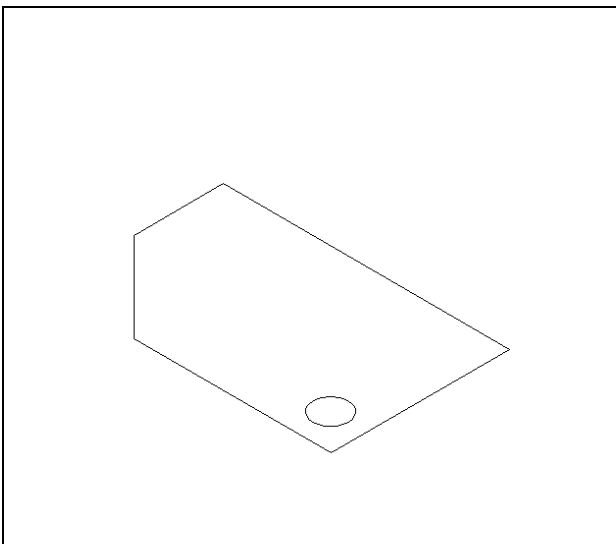


Figure 8.11 – Southwest View

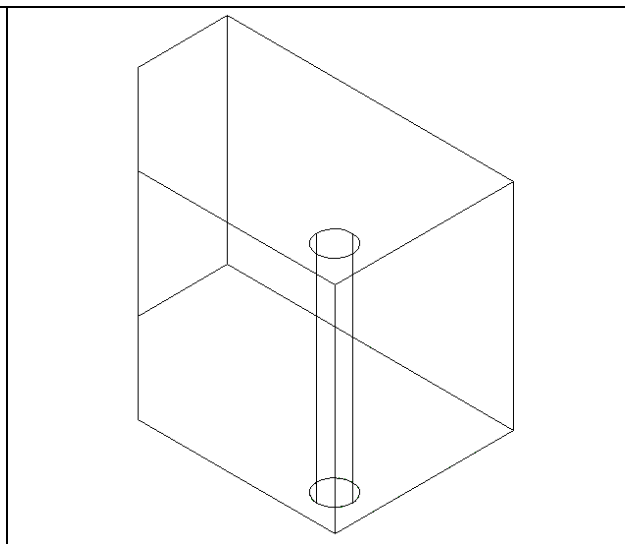
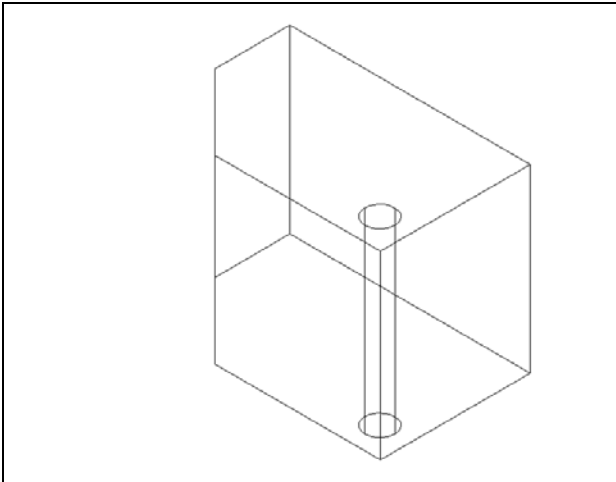


Figure 8.12 – Extruding Two Entities

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane with the vertical Z-axis rising vertically as shown in Figure 8.11. Select the Extrude tool on the Solids toolbar and the command line will prompt you to “**Select entities:**”. Use a selection window to select both shapes. The command line will return with “**Entities in set: 2**”. Hit **ENTER** to specify the height or path of the solid. Type “**3.0**” for the height of the extrusion, hitting **ENTER** again to keep the angle of taper at 0 degrees. The shapes will extend 3 units up into the z axis as shown in Figure 8.12.

Subtracting a 3D Solid from Another Solid

To remove the cylinder from the master solid, select the Subtract tool on the Solids Editing toolbar.



The command line will prompt you to “**Select ACIS object to subtract from:**”. Select the larger master solid and the system will respond with “**Entities in set: 1**”. Hit **ENTER** to proceed to the next step, which is to “**Select ACIS objects to subtract:**”. Pick the cylinder and the software application will counter with “**Entities in set: 1**”. Hit **ENTER** and a region exactly the shape of the cylinder will be removed from the main solid as shown in Figure 8.13. Notice that you cannot see any difference in the solid after subtracting the cylinder.

Figure 8.13 – Subtracting the Cylinder

Rotating a 3D Solid Using the Rotate3D Command

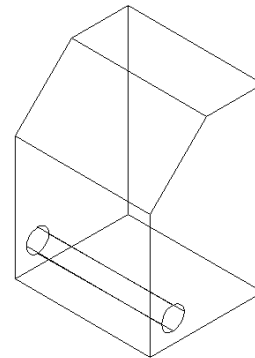
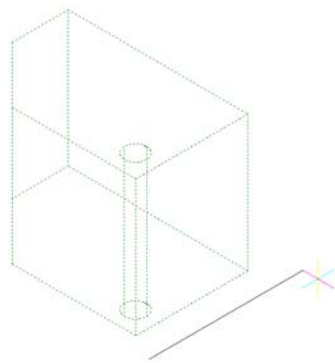
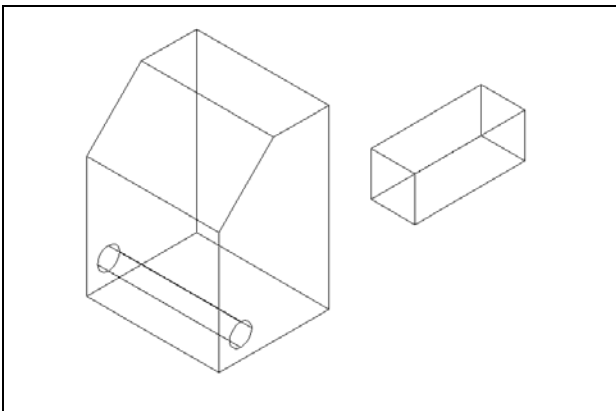


Figure 8.14 – Selecting the Axis of Rotation Figure 8.15 – The Rotated Solid

You can rotate the solid to the left on the X-axis using the Rotate3D command. Select Modify from the Menu Bar, then 3D Operations, and pick Rotate 3D from the list of commands. At the command, “Select entities to rotate:”, pick the solid and hit **ENTER** to proceed to the second part of the function. You are going to rotate the solid part on the x-axis, so pick any point on the display and then with the Ortho mode still “on” pick a second point to draw a line on the x-axis. Type **90** to rotate the solid as shown in Figure 7.32.

Drawing a Solid Box

The technique you will use in this exercise will utilize the Length option of the Box command.



Select the Box tool on the Solids toolbar. At the command line, a prompt asks you to specify the corner of the solid. For this box, specify the point anywhere to the right of the solid. Then type **L** after specifying the starting point and the system will ask you three questions. Specify the “Length of side of box:” as **2.5** for the X coordinate. Enter **1** as the value for the width and height of the box. The rectangular solid will appear as shown in Figure 8.16.

Figure 8.16 – Placing a Solid Box

Moving a 3D Solid

The Move command is actually a three-dimensional function in progeCAD Professional. The way you can determine whether a command is three dimensional in nature is to attempt to use the function outside the Z-plane. You can use move to relocate any entity to another point (X,Y,Z) in Model Space.

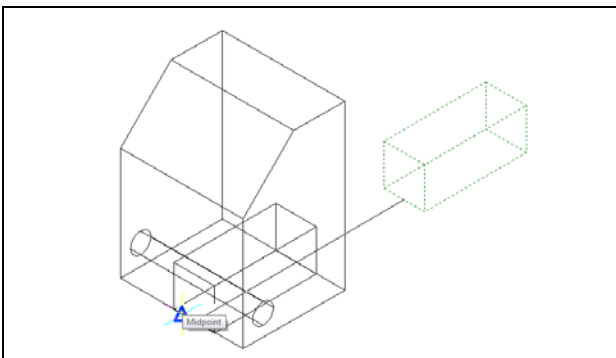


Figure 8.17 – Moving the Box

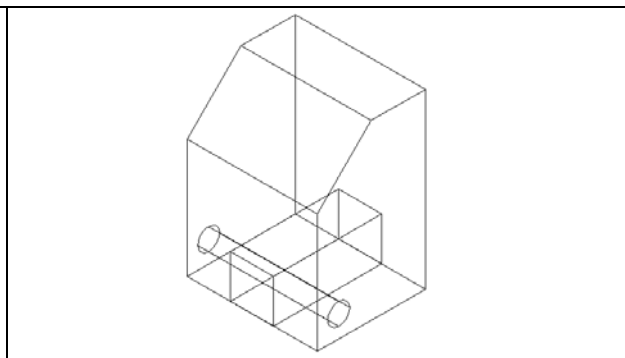


Figure 8.18 – The Moved Box

To move the box, select the Move tool on the Modify toolbar. Once you choose the Move command, the prompt will be to “Select entities to move:”. Pick the solid box and the command line will respond with “Entities in set: 1”, so just hit **ENTER** to go the second part of the command. Next, you need to specify a “Vector/<Base point>:” on the graphical display. Pick your base point with the Midpoint Esnap, selecting the Midpoint of the bottom left edge of the box as shown in Figure 8.17. The “Displacement point:” is the midpoint of the bottom left edge of the master solid. Pick the second point using the Midpoint Esnap and the solid box will move as shown in Figure 8.18.

Constructing a Solid Perimeter Using the Line Command

You need to create a right triangle with a 2-inch hypotenuse. You may want to be in the top orthographic view to draw the three-sided shape. After changing your view, select the Line tool from the Draw toolbar.



Again, you will let the computer to do the mathematical work to create the triangle. Pick the starting point to the left of the solid and with the Ortho “On”, draw a line 2.0 to the left. Now you will draw a line using polar displacement. Type **@2.0<-45** and a line will appear as shown in Figure 8.19. Hit **ENTER** to exit the Line command.

As before using relative coordinates, the “@” symbol moves the entity from the previous point. The 2.0 is the distance required to move and the “-45” is the angle in degrees from the base of “0” degrees. In progeCAD, the zero degree base angle is in the right or positive X-axis. The “-45” angle draws the line 45 degrees to the bottom of the 2.0 line.

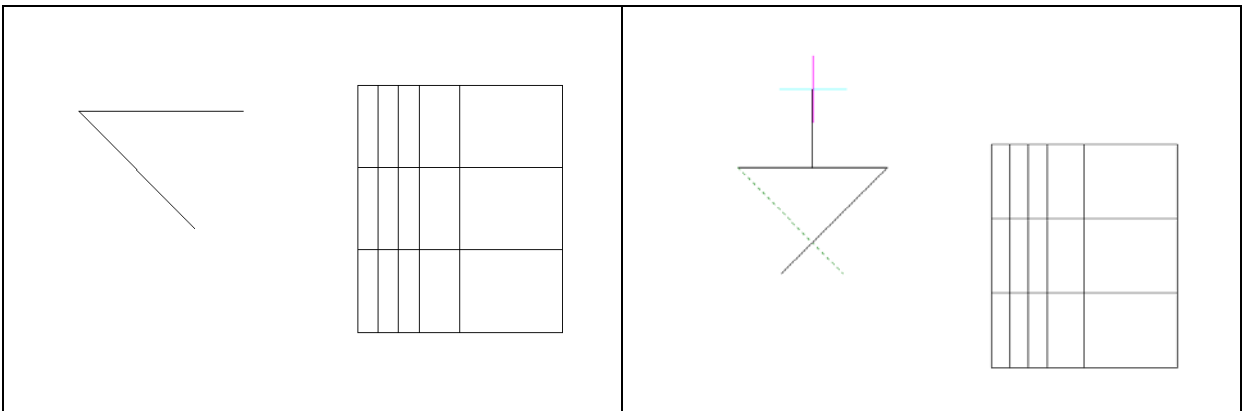


Figure 8.19 – Drawing the Line

Figure 8.20 – Mirroring a Line

You will use the Mirror tool on the Modify toolbar to copy the 45-degree line across the midpoint of the 2.0 line. At the prompt to select entities to mirror, pick the angled line and then hit **ENTER** to proceed to the second half of the Mirror command. Specify the “Start of mirror line:” by picking the Midpoint Esnap of the horizontal line and then select the “End of mirror line:” by picking just above the first point with the Ortho “On” as shown in Figure

8.20. The default to delete the source objects is no, so at the prompt “Delete the original entities? <N>:” just hit **ENTER** and the mirrored and duplicated line from will appear on the other end of the horizontal line (Figure 8.17).

Pick the Fillet tool on the Modify toolbar, type “**R**” for radius, “**0**” for the radius value and then hit **ENTER**. Select the two 45 degree lines and the intersection of the 45-degree triangle will become a zero radius. You can either use the Fillet or Trim tool to create the triangle as shown in Figure 8.22.

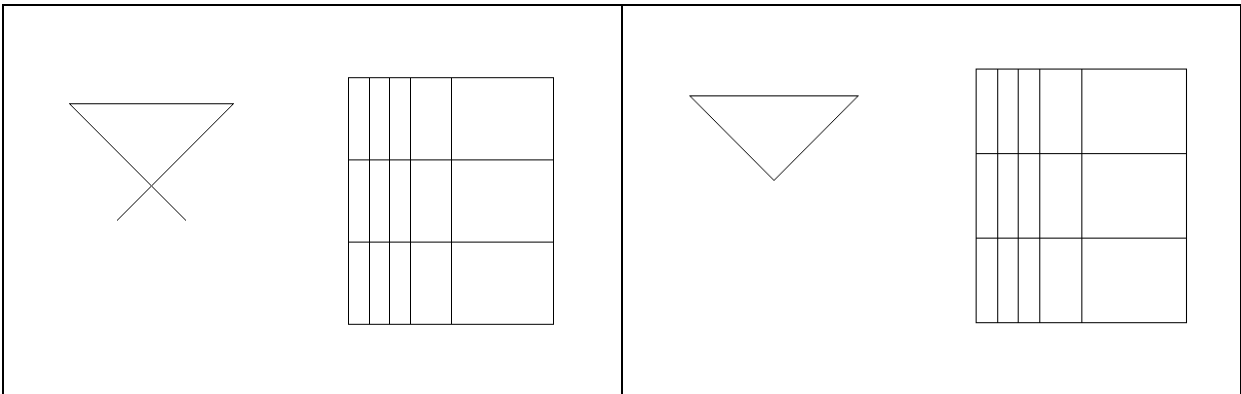


Figure 8.21 – Fillet the Lines, Radius 0

Figure 8.22 – The Finished Triangle

Creating Polylines Using the Edit Polyline Tool

To merge the three lines segment together you need to use the Edit Polyline tool. Select the Edit Polyline tool on the Modify II toolbar, and then select one of the objects.

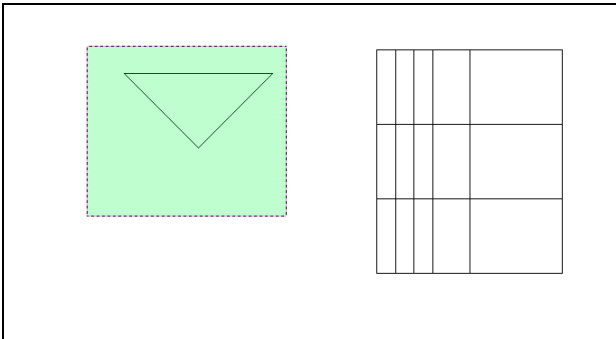


Figure 8.23 – Polyline Edit the Right Side

Type J and ENTER to go to the Join feature of the Edit Polyline tool. You will be prompted to “Select entities to join.” Use a selection window to select all of the lines in the triangle and hit ENTER. The command line will respond with “Polyline is now closed. 1 vertex added to polyline.” Hit Escape to exit the Edit Polyline command.

Extruding a Solid from Closed Polyline

Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar. Change to the Southwest Isometric view to watch your extrusion happen in realtime.

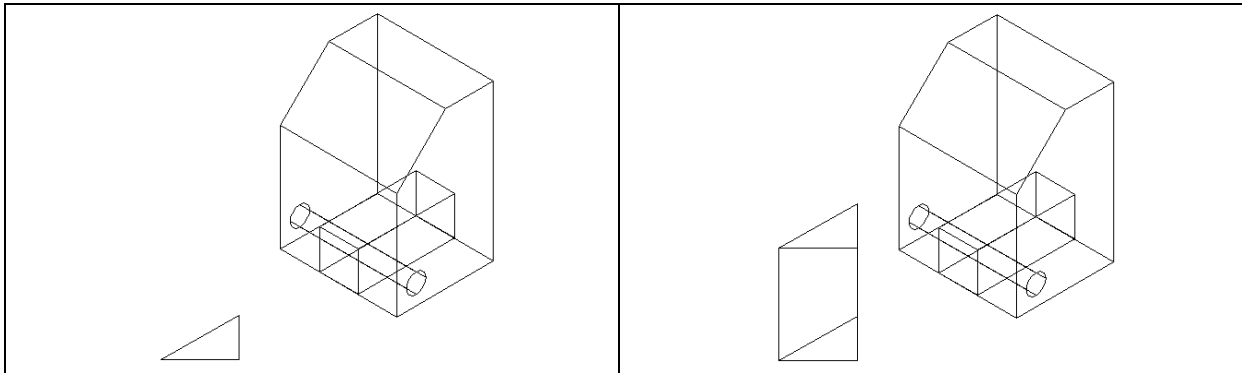


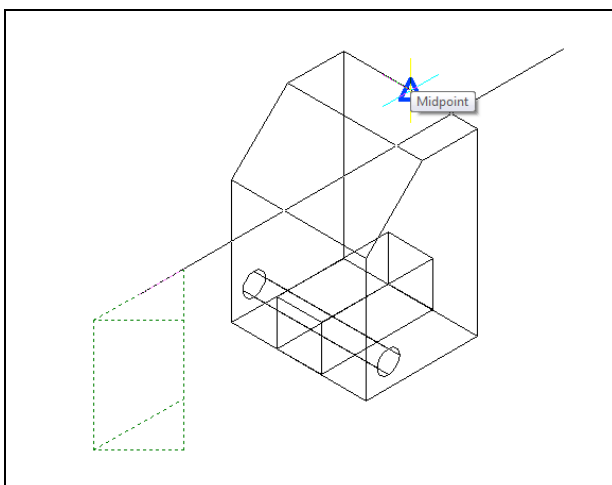
Figure 8.24– Southwest Isometric View

Figure 8.25 – The Extruded Triangle

Select the Extrude tool on the Solids toolbar and the command line will prompt you to “**Select entities:**”. Use a selection window to select the triangle. The command line will return with “**Entities in set: 1**”. Hit **ENTER** to specify the height or path of the solid. Type “**2.5**” for the height of the extrusion, hitting **ENTER** again to keep the angle of taper at 0 degrees. The triangle will extend 2.5 units up into the z axis as shown in Figure 8.25.

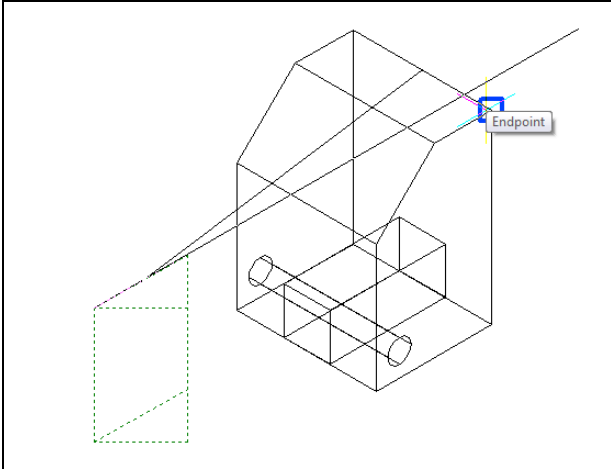
Aligning a Solid to the Master Solid

Select Modify on the Menu Bar, then 3D Operations, and pick Align from the list of commands. At the command, “Select entities:”, pick the triangular solid and hit **ENTER** to proceed to the second part of the function.



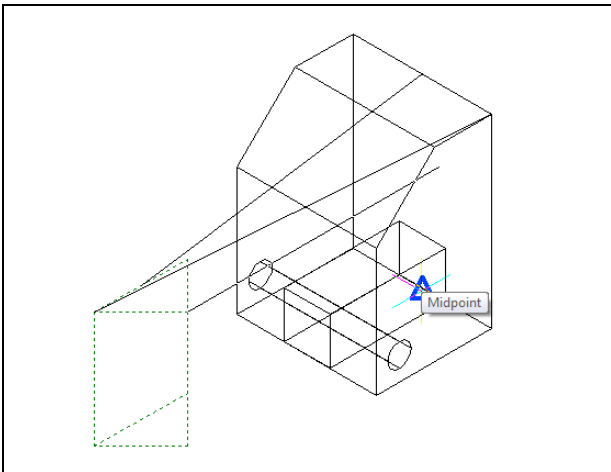
The command line will ask you to “Specify first source point:”. Select the midpoint Esnap of the hypotenuse of the triangle that is the top face of the solid. When the command line prompts you to “Specify first destination point:” select the midpoint of the top back edge of the master solid as shown in Figure 8.26. In the Align command, the first source point will always join with the first destination point so that the two will share the same coordinate. A temporary line will appear to show the relationship between the two points.

Figure 8.26 – The First Alignment Points



Next the command line will prompt you to select the second source and destination points. With the second and third sets of point selections you will designate the plane on which your two solids will become aligned. Select the second source point as the left-hand corner of the top of the triangular solid, and then the second destination point as the upper right-hand corner of the of the master solid. A temporary line will appear to designate the relationship between the two points (Figure 8.27).

Figure 8.27 – The Second Alignment Points



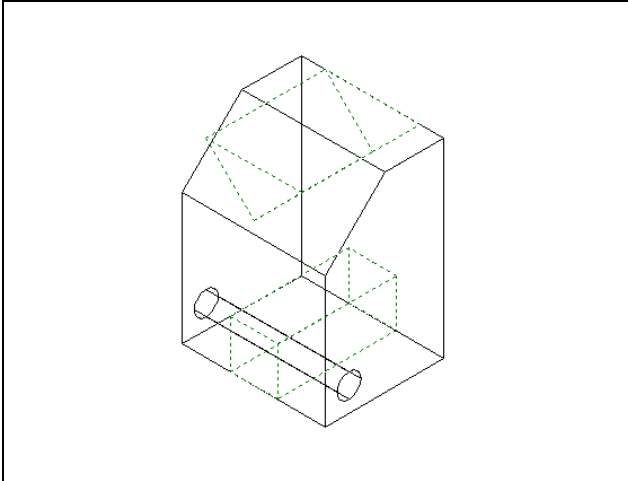
Finally you are prompted to select the third source and destination points. Choose the top 9 degree angle point as the as the source point on the triangular solid and the midpoint Esnap of the bottom back edge of the master solid as the destination point. Immediately the source part will move to the chosen alignment as shown in Figure 8.29.

Figure 8.28 – The Third Alignment Points

Subtracting 3D Solids from the Master Solid

To remove the two remaining solid details from the master solid, select the Subtract tool on the Solids Editing toolbar.



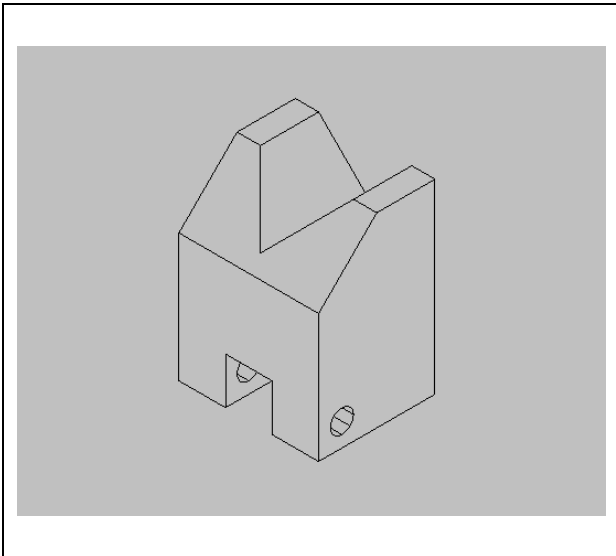


The command line will prompt you to “Select ACIS object to subtract from:”. Select the larger master solid and the system will respond with “Entities in set: 1”. Hit **ENTER** to proceed to the next step, which is to “Select ACIS objects to subtract:”. Pick the box and the triangular solid and the software application will counter with “Entites in set: 2”. Hit **ENTER** and regions exactly the shape of the selected solids will be removed from the main solid as shown in Figure 8.30. You will want to use the Hide command to see your results better.

Figure 8.29 – Subtract the Solids

Hide the 3D Solid

In Figure 8.29, you will notice that the tessellation lines representing the solid’s outline on the far edge of the part are visible. In a real world solid part, you would not be able to see through solid unless its material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.



Select the Hide tool on the Shade toolbar. This is a very simple command because after picking the tool the tessellation lines representing the back of the part are removed from view as shown in Figure 8.30. If you wish to return the graphical display to the previous setup in which all the tessellation lines can be seen, choose the first tool on the Shade toolbar called 2D Wireframe. By choosing the 2D Wireframe tool, the solid part and the UCS icon will return to the previous appearance. This command does not alter the mass of the solid or any of the dimensions, but is available for the computer aided designer as a viewing option.

Figure 8.30 – Hide the Solid

Moving the Solid to the Origin Point

To move Problem Seven to the drawing origin of 0,0,0 on the x, y, and z axis, pick the Move tool on the Modify toolbar, select the 3D solid, and hit ENTER. When prompted to select a “**Vector/<Base point>:**,” select the endpoint shown in Figure 8.31 at the lower left-hand side of the Problem Seven, which will become its insertion point if it were inserted into an assembly drawing. For the prompt to select a “Displacement point:,” type **0,0,0** and hit ENTER. Problem Seven will move to the new origin point. Try using Zoom Extents if the part completely escapes your viewing area.

*** World Class CAD Challenge 101-07 * - Close this drawing file. Create a New file and draw the solid using the techniques in this chapter. Move the finished solid to the origin of the drawing. Complete the task in less than 5 minutes. Continue this drill four times, each time completing the drawing under 5 minutes to maintain your World Class ranking.**

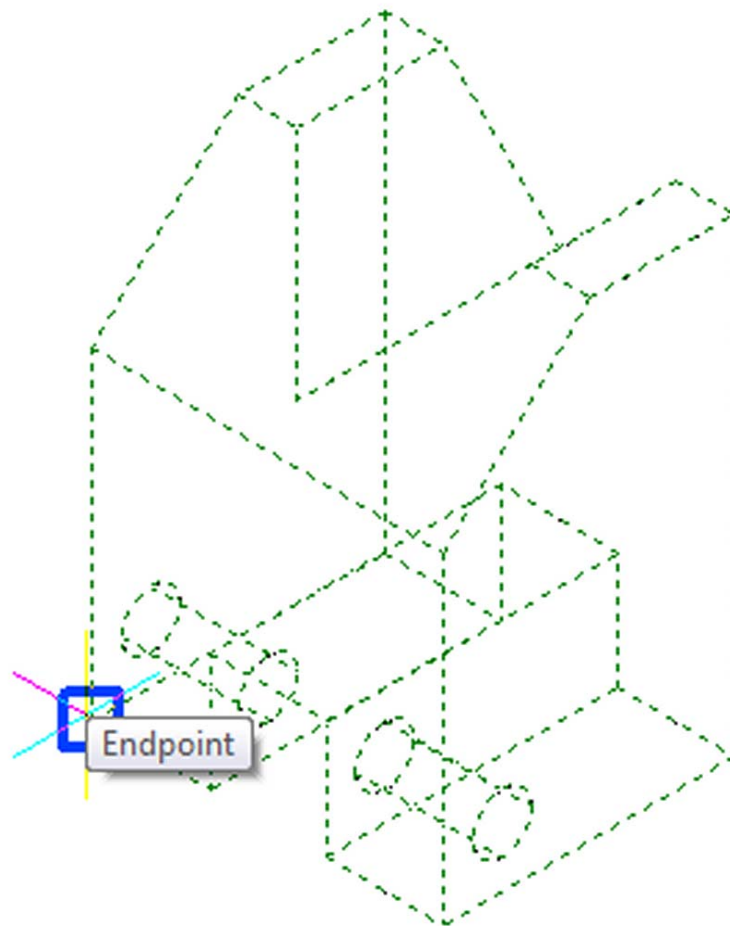


Figure 8.31 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem Seven, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type “Problem 7” and press the Save button to save the drawing (Figure 8.32).

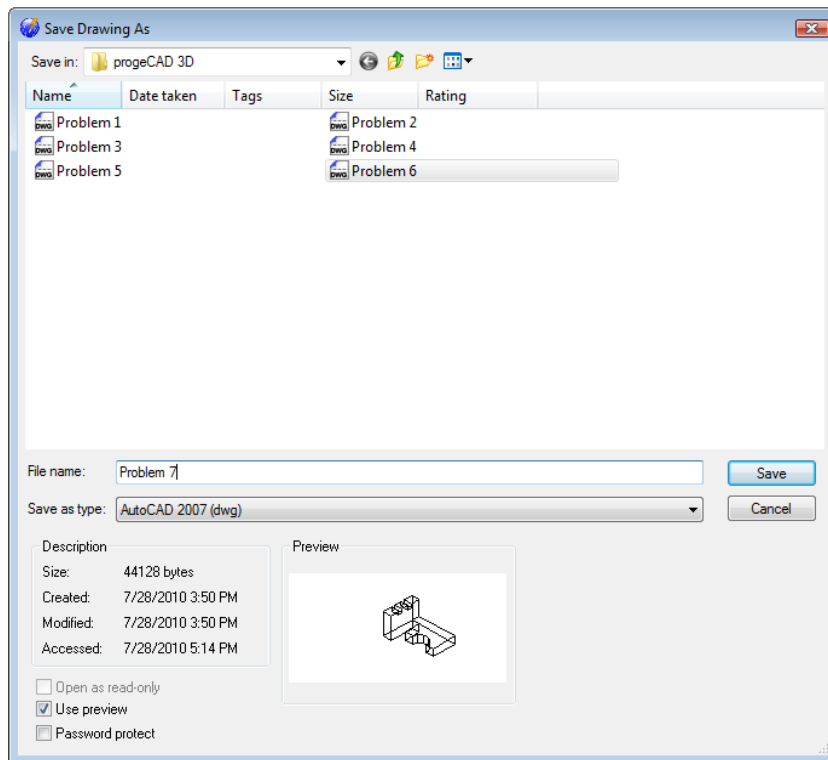


Figure 8.32 – Saving Problem Seven

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**